

## CROSS-REFERENCE TO RELATED APPLICATIONS

This application claims priority under 35 U.S.C. §119 to European patent application No. 99-117-379.0, filed September 3, 1999, by Peter Hummel, Igor Jacak, and Stanislav Hutnan, and entitled "DEFINING PARAMETERS FOR AN FEA CALCULATION IN A CAD PROGRAM," which application is incorporated by reference herein.

## 1. Field of the Invention

The present invention concerns the fields of computer aided design (CAD) and finite elements analysis (FEA). In particular, the present invention concerns a way of improving the integration of these fields into a unified design environment.

The use of computer aided design (CAD) techniques has become common engineering practice. The available CAD programs range from simple drawing tools to sophisticated systems covering the whole range of product design and possibly further aspects like engineering or manufacturing or quality control. The term "CAD program" as used herein should therefore be understood in its broadest meaning as any computer program that contains a drawing and/or design component and possibly further components. A widely used CAD program is manufactured in various versions by Autodesk, Inc., San Rafael, USA under the trademark AutoCAD.

For many available FEA programs, the definition of the input parameters is a rather cumbersome process. While it is normally possible to import data defining the body to be analyzed from a CAD program, load and support parameters must be defined in a complex way. In other words, there is often too little compatibility and integration between independent CAD and FEA programs.

25 A CAD program with integrated FEA functions is currently available under the trademark Genius Desktop 3 from Autodesk, Inc., San Rafael, USA. This program makes it possible to perform FEA for any three-dimensional body defined using the underlying CAD functionality. The results of the FEA calculations are also displayed for the user in a fully integrated way. However, the Genius Desktop 3  
30 product imposes certain limitations with respect to the definition of load and support parameters (load/support conditions). It is only possible to define forces and fixed and movable supports that act on individual points or along edges or on

whole faces (delimited by edges) of the body. These limitations restrict the variety of situations that can be modeled conveniently. Any attempt of the user to work around these limitations would be rather complex and time-consuming, and would furthermore decrease the accuracy of the finite elements analysis and increase the necessary computing time.

Therefore a need exists for improving the integration between CAD programs and FEA programs or program modules. In particular, a need exists for expanding the possibilities for defining parameters like load or support conditions for an FEA calculation in a CAD program. The invention has the objective of meeting these needs fully or at least partially.

#### SUMMARY OF THE INVENTION

The present invention provides a method for defining at least one parameter for an FEA calculation in a CAD program as defined in claim 1, a computer program product having the features of claim 16, and an apparatus having the features of claim 20. The dependent claims concern preferred embodiments of the invention.

A basic idea of the present invention is that at least one graphical function of the CAD program is used to define a region within a face of a body for which an FEA parameter is to be entered. This region is then used to define a load/support condition for the FEA calculation. For example, it may be defined that a fixed or movable support or a load acts on the defined region or on the portion of the face outside the defined region. In this respect, the wording "region within a face" means that the present invention makes it possible for the user to define regions that do not comprise the whole face. A "face" of a body is generally understood (as defined in the boundary representation model) to be a portion of the surface of the body delimited by edges, or the whole surface of the body if there are no edges (e.g. for spheres or tori). A "load/support condition" is a condition or parameter defining a load (force) acting on a body and/or a support of the body.

By allowing the user to define load/support conditions with respect to regions of a face of the body, the present invention opens a new class of possibilities for finite elements modeling. Consider the example that a user wants to model the mechanical properties of a table on which a drinking glass has been placed.

- 5 Clearly, the weight of the glass only acts on the (usually circular) region where the glass contacts the table surface. Using the present invention, this situation can be modeled very conveniently by simply taking the region where the drinking glass is in contact with the table surface as the region used for defining the load/support condition. This region can be defined in a natural way using the functions  
10 provided by the CAD program.

- Generally, all types of CAD functions and operations can be used for defining the region within the face of the object. This may include, in preferred embodiments, functions for drawing an object on the face of the body, and/or functions for  
15 drawing an object and projecting it onto the face of the body, and/or functions for subtracting (intersecting) the body from another object or body, and/or functions for selecting existing objects, and/or other functions. Especially in the context of powerful CAD programs implementing a wide variety of graphical functions, the range of possibilities is increased greatly. The face and/or the region may be flat  
20 or straight or curved.

- The definition of one or more parameter/s for an FEA calculation in a CAD program is possible either in a way that FEA functions are tightly integrated into the CAD program, or in a way that only the parameter value/s is/are obtained in  
25 the CAD program and stored or output for use by an external FEA program. Both these extremes as well as intermediate possibilities (e.g., having a CAD and an FEA program running in parallel and communicating via operating system services) are contemplated for implementing the present invention. In the terminology used herein, an "FEA calculation" comprises all FEA steps including  
30 the initial mesh generation and subsequent calculation steps.

09651031-083000

5

10

15

20

## 25

30

Fig. 1 shows a block diagram of the program structure used for implementing the present sample embodiment,

Fig. 2 shows a flow diagram of the context in which the present sample embodiment is embedded,

Fig. 3 and Fig. 4 depict possible dialog windows that are shown to the user at  
5 certain processing stages of the present sample embodiment,

Fig. 5 shows a flow diagram of the method of defining a region within a face in the present sample embodiment,

10 Fig. 6 shows a body as displayed during face selection,

Fig. 7a - Fig. 7c depict possible ways in which a body is displayed during the method of defining a region within a face in the present sample embodiment,

15 Fig. 8 shows a flow diagram of some preparatory steps of the finite element analysis performed in the present sample embodiment, and

Fig. 9a - Fig. 9c illustrate some of the steps shown in Fig. 8.

20 DETAILED DESCRIPTION OF THE PREFERRED EMBODIMENT

The present sample embodiment is implemented as a CAD program 10 (see Fig. 1). The CAD program 10 may run on a powerful PC type workstation having an operating system 12 like one of the operating systems known under the  
25 trademarks Windows 98 or Windows NT, available from Microsoft Inc., Redmond, USA. The CAD program 10 comprises a CAD core program 14 and CAD extensions 16, the CAD extensions 16 in turn containing an FEA module 18 for three-dimensional FEA calculations. In the present sample embodiment, the CAD core program 14 is the product known under the trademark AutoCAD 2000,  
30 available from Autodesk, Inc., San Rafael, USA, and the CAD extensions 16 are extensions for mechanical engineering design.

5

15

20

25

30

5

10

15

30



enter further properties of the load/support condition in step 44. For example, such further properties are the amount and direction of load forces and the direction of movable support forces. For load and movable support forces that act on a curved loop or face or region, it must be defined whether the direction of the force is constant or normal to each point of the loop/face/region. Furthermore, the definition of a region within the face F may either mean that the corresponding load and support forces act inside the region, or within the face F, but outside the region. This information may be provided to the CAD program either during step 42 (e.g., by positioning the insertion point used when defining the region inside or outside the region), or in step 44. Further or fewer or other properties of the load/support conditions may be required in alternative embodiments.

Step 44 finishes the definition of a single load/support condition in the presently described embodiment. The user may now enter another load/support condition in the same way as described above ("yes" branch of test 46 jumping back to step 22). When all load/support conditions have been defined ("no" branch of test 46), properties of the material of the body B, on which the finite element analysis shall be based, are entered in step 48. The user may directly enter physical parameters like the modulus of elasticity or Young's modulus, the yield point, the Poisson's ratio and the density of the material, or he/she may take these parameters from a predefined table (see group box "material" in the main dialog window of Fig. 3).

Referring again to Fig. 2, the FEA is now performed in step 50, and the results are displayed in step 52. A more detailed description of step 50 will be given below in connection with Fig. 8. The various possibilities for and methods of displaying the results to the user in step 52 are known in the art of finite element analysis and are not the subject of the present invention.

The main dialog window shown in Fig. 3 comprises control fields for the individual steps and operations mentioned above. The control fields showing icons in the

top group box (entitled "loads and supports") concern the selection of step 22. From left to right, these control fields define:

- a load acting on a single point,
- 5    - a fixed support acting on a single point,
- a movable support acting on a single point,
- a uniform load acting along a loop,
- a fixed support acting along a loop,
- a movable support acting along a loop,
- 10   - a uniform load acting on a whole face or a region within the face,
- a fixed support acting on a whole face or a region within the face, and
- a movable support acting on a whole face or a region within the face.

The further group boxes in the main dialog window of Fig. 3 concern the definition of material properties (group box "material" corresponding to step 48 of Fig. 2), the running and refining of FEA calculations (group boxes "run calculation" and "refining" corresponding to step 50 of Fig. 2), and the various possibilities of displaying the results (group box "results" corresponding to step 52 of Fig. 2).

20 The flow diagram of Fig. 5 shows the methods available for performing step 42 in Fig. 2, i.e., for defining a region within the selected face F of the body B. As mentioned above, the dialog window shown in Fig. 4 is displayed to the user after selection of the face F in step 38 of Fig. 2. A mouse click on one of the six selection fields in the group boxes "select existing" and "draw and project on the

25 face" initiates the corresponding method for selection and, if applicable, also defines the type of an object to be drawn (step 54 in Fig. 5).

The "yes" branch of test 56 is taken if the user clicks on the control field "entity" in the group box "select existing" of the dialog window (Fig. 4). In this case, an existing object is chosen by the user for defining the region of the load/support condition (step 58). The selected object may, for example, be a circle or a closed polyline or a rectangle. In different embodiments of the present invention, it may

or may not be required that the selected object is fully or partially contained within the face F.

A second way for defining the region within the face on the basis of existing  
 5 objects can be chosen by clicking on the control field titled "body for subtraction" in the group box "select existing" of the dialog window (Fig. 4). In this selection method ("yes" branch of test 60 in Fig. 5), the user first chooses another existing body (step 62) intersecting the body B selected in step 20. The region used for defining the load/support condition is then delimited by a loop of the intersection  
 10 of the newly selected body with the face F (selected in step 38) of the body B. First, in step 63, the intersection is calculated by a subtraction operation, and all loops of the intersection that are fully contained within the selected face F are determined. If there is only one such loop ("yes" branch of test 64), it defines the region within the face F. If there are several loops ("no" branch of test 64; this  
 15 may happen if, for example, the selected face F is spherical or cylindrical), the user is prompted to select one of these loops for defining the region (step 65).

The method "select existing body for subtraction" may also be used for determining regions defined by the contact area between the face F selected in  
 20 step 38 and the body selected in step 62. Therefore, this method would be appropriate in the example mentioned initially where a drinking glass standing on a table was to be modeled.

The three control fields named "rectangle", "polyline" and "circle" in the group box  
 25 "draw and project on the face" of the dialog window of Fig. 4 each call a method for drawing an object. As an example, Fig. 7c shows a square object O being drawn on the upper face F of the selected body B. In this example, the object O exactly corresponds to the region R to be defined within the face F. This is because both the object O and the face F are flat and because the object O is  
 30 fully contained in the face F.

000000" T00T5960

5

15

25

30

5

10

20

30

5

10

15

25

30

- 10 and overlapping properties of the regions R1, R2.

20 is used.

Fig. 9b), while two-sided front lines will be generated for the border portions of the regions R1, R2 (double-headed arrows in Fig. 9b).

A mesh generator, which is known per se in the art, is now called in step 100 to generate a triangular meshing for the whole face F. The mesh is generated such that each of the initial front lines obtained in step 98 coincides with a side of a mesh triangle. This ensures that the generated mesh is compatible with the geometrical shapes of the defined load/support conditions. Fig. 9c shows, as an example, the meshing generated for the inside of the region R2. The meshing outside of the region R2 has also been generated in step 100, but it is not shown in Fig. 9c for the sake of clarity.

In step 102 the load conditions for each triangle of the mesh are determined. For each main symbol inserted by the user ("x"-shaped mark in Fig. 9a - Fig. 9c), the following operations are performed: the mesh triangle enclosing the main symbol is identified, and all bordering triangles are recursively determined until a border of the corresponding load/support condition is reached. Thus all triangles for which a load/support condition applies are identified. Of course, there may be triangles for which several load/support conditions are applicable (e.g., the triangles in the intersection portion of regions R1, R2 in Fig. 9c). For example, if a triangle is covered by more than one load condition, the sum of the forces defined in these conditions will be taken as the force acting on the triangle.

This completes the preparation and meshing steps. The remaining FEA calculations are now performed in step 104. These calculations are known in the art (e.g., from the books mentioned initially) and are not the subject of the present invention.